Computers & Fluids 96 (2014) 288–301

Contents lists available at ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

MRT lattice Boltzmann method for 2D flows in curvilinear coordinates

Ljubomir Budinski *

Faculty of Technical Sciences, University of Novi Sad, Trg Dositeja Obradovica 6, 21000 Novi Sad, Serbia

ARTICLE INFO

Article history:
Received 11 February 2013
Received in revised form 6 January 2014
Accepted 11 March 2014
Available online 1 April 2014

Keywords:
MRT lattice Boltzmann method
Curvilinear coordinates
Complex geometry
Shallow water equations
Navier–Stokes equations

ABSTRACT

Objective: The objective of the presented paper is establishing the multi-relaxation-time lattice Boltzmann method (MRT-LBM) for solving 2D flow equations transformed in a curvilinear coordinate system.

Method: Using the complete transformation approach – which includes transformation of both dependent and independent variables between the physical and computational domain – corresponding forms of the equilibrium function and of the force term for the 2D Navier–Stokes equations and the shallow water equations have been derived. The physical flow domain of arbitrary geometry in the horizontal plane, is covered with adequate curvilinear mesh, while the calculation procedure is carried out in the D2Q9 square lattice, applying the basic form of the boundary condition method on water-solid and open boundaries as well.

Test cases: The method is tested using four different examples: **Couette** flow in a straight inclined channel, Taylor–Couette flow between two cylinders, a non-prismatic channel in a 180° bend, and a segment of irrigation channel with a parabolic cross section in a 90° bend. In the cases of the bent channels, previously available velocity measurements have been used for validation of the model. In addition, the procedure employs a mathematical model based on traditional CFD procedures.

Results: The remarkable agreement between the results obtained by the proposed model and the corresponding analytical values and measurements shows that the presented curvilinear form of the LBM is capable of solving very complex environmental problems, maintaining the order of accuracy, simplicity and efficiency of the basic LBM.

1. Introduction

Based on the kinetic structure Lattice Gas Automata (LGA) [1,2], the lattice Boltzmann method (LBM) represents a new approach for modeling governing equations of Computational Fluid Dynamics (CFD). Solving particular forms of fluid motion equations indirectly (Navier–Stokes equations, Reynolds equations) – wherein, based on principles of statistical physics [3] and the kinetic theory of matter – the motion of particles is modeled on a mesoscopic level, resulting in a very powerful numerical procedure for solving complex physical processes related to fluid motion (multiphase flow, flow in porous media). Therefore, the area of application of the LBM increases (river hydraulics, chemical industry), together with further improvements in the actual basic structure of the LBM. These improvements enable the LBM to solve a wide range of a complex flows, characterized by a variety of boundary conditions. Accordingly, a crucial element of modeling these kinds of flows, challenging the essential numerical characteristics of the LBM, is the correct and adequate definition of boundary conditions characteristic of domains of arbitrary shape.

CFD, recognized as a scientific discipline aimed at defining and studying relevant physical processes related to "real physical states and conditions" of fluid motion, first of all, requires an adequate and accurate description of complex hydraulic conditions. Since the main cause of hydraulic complexity lies primarily in the geometrical structure of the referred water body, a huge portion of research in the area of CFD is aimed at producing and improving existing numerical procedures capable of coping with problems imposed by geometrically complex domains. Compared to the classic Computational Fluid Dynamics (finite difference, finite element method) the LBM represents a relatively recent approach in the field of CFD. To start with, an appropriate method for solving fluid flows in domains of arbitrary geometry had to be adopted. Several different approaches have been developed for this purpose in the past two decades. To overcome difficulties in applying boundary conditions along boundaries not coinciding with the direction of the main axes, Filippova and Hänel [4] proposed a procedure of local hierarchical grid refinement on the classic uniform calculation grid. This approach has been widely used and further improved by many researches [5–9]. In an attempt to increase

* Tel.: +381 658083688.
E-mail addresses: ljubab@open.telekom.rs, ljubab@gf.uns.ac.rs

http://dx.doi.org/10.1016/j.compfluid.2014.03.007
0045-7930/

capability of the LBM in solving flows in domains of arbitrary geometry, authors He et al. [10] first applied the interpolation-supplemented scheme (ISLBE). Using non-uniform computational grids and the Lagrangian nature of the LBM, the computational domain is covered with a body-fitted rectangular grid, while the unknown distribution functions are calculated using interpolation techniques [11–13]. Furthermore, to improve efficiency of the proposed method, Shu et al. [14] recently introduced the Taylor series expansion and optimization by the least squares method for determining the distribution functions. The application of this method, which can be seen as an extended and enhanced form of the interpolation-supplemented LBM, can be found in [15–17]. Since all proposed configurations of the LBM for non-uniform grids introduce additional steps in the calculation process – which in a certain sense affects the basic nature of the method – another approach has been developed in the domain of rectangular grids. In order to preserve computational efficiency of the basic form of the LBM in cases of non-uniform grids, Zhou [19] has introduced a new form of local equilibrium distribution functions; while Bouzidi et al. [18] have proposed a unique form of the transformation matrix for the multiple relaxation time model (MRT). Besides the above-mentioned procedures, models based on discretization of the partial differential Boltzmann equation transformed in curvilinear coordinate system utilizing classic CFD techniques, are also available [20–23].

Application of the proposed variety of LBMs on non-uniform rectangular grids refers to the domain within the boundaries. However, additional techniques are required in the neighborhood of the solid–water boundary. In case of arbitrary geometry, when the boundary line intersects with the calculation grid, the existing boundary condition methods (bounce-back, elastic-collision scheme) need to be modified. The first attempt to solve the unknown distribution function "coming" from a "rigid" node to the "fluid" node using a prescribed formulation and interpolation technique was made by Ginzburg and Adler [48]. Filippova and Hänel [4] proposed a procedure for application on optionally bent boundaries stretching between the nodes of the lattice. Improvement of the proposed method in terms of stability and efficiency is outlined in [24–28]. Another approach implying reconstruction of the hydrodynamic variables using density, velocity and rate of strain, instead of the distribution function, originates from Lätt et al. [29]. It was subsequently improved by Verschaeve and Müller [30]. However, additional interpolation/extrapolation operations have adverse effects on the proposed methods compared to the basic form of the LBM in terms of efficiency, simplicity and accuracy. In addition to the methods outlined above, certain authors overbridge the problem of boundary conditions imposed by complex geometry using discretization (finite difference, finite volume method) of the partial differential Boltzmann equation [22].

Generally, in dealing with domains of arbitrary geometry, CFD requires supplementary steps, regardless of whether the classical method of discretization of the flow equations or the LBM approach is used. In case of traditional CFD methods, writing flow equations in curvilinear coordinates results in bulky expressions. For this, the efficiency and accuracy of the methods available for the discretization and solving of the obtained algebraic equations become significantly deteriorated. On the other hand, as previously mentioned, introducing additional steps in the LBM is the main disadvantage of the proposed techniques, since they cause a loss of the original advantageous nature of the LBM. In order to maintain the simplicity and efficiency of the LBM in cases of domains with complex geometry, the option of avoiding additional terms together with forcing the basic, straightforward forms of boundary conditions is explored by bringing together the LBM with methods established in the classic CFD (finite difference) [31]. Using a complete transformation of the 2D flow equations in curvilinear coordinates and covering the area of interest with a body-fitted mesh provide appropriate equilibrium distribution functions and force terms. Then, analogous to the finite-difference methods (FDM), a square lattice is used for the computational grid. Since the transformed flow equations now have contravariants as dependent variables, as opposed to velocity components in the Cartesian coordinate system, application of the boundary conditions is restored to its basic form, while the simplicity of the method as well as its suitability for parallelization is maintained. By this approach, demand for mapping the locations and defining the "nature" of the boundary intersections is eliminated, which in turn generalizes the process of flow modeling in domains of complex geometry.

To enhance the stability of the proposed curvilinear form of the LBGK (lattice Boltzmann Bhatnagar-Gross-Krook), which is the main weakness of the LBM in cases of high Reynolds numbers characteristic to natural and artificial watercourses, the multi relaxation time method (MRT) for the curvilinear LBM is introduced in this paper. Using a technique whereby the relaxation of the non-conserved moments towards the equilibrium in the moment space is performed with different relaxation times, the stability of the method in the domain of high Reynolds numbers is significantly increased. The basic theory and the corresponding applications of the MRT method are available in [32–37]. Since modeling most of the flows of interest exceeds the capability of the single relaxation time LBGK method [38], the MRT method enhanced in sense of simulation flows of high Reynolds numbers in domains of complex geometry, now yields the opportunity of modeling flows in natural water bodies. This has been tested by four examples in this paper. The first test is carried out on a straight channel set inclined to the main axis in baseplot. The second example is a circular Couette flow between two coaxial cylinders. The third and fourth examples analyze a prismatic flume in a 90° and 180° bend, respectively. The numerical results of the straight channel are compared to the corresponding analytic solution, whereas velocity and depth measurements are used for the validation of the proposed method in the cases of the 90° and 180° bends. Applying standard error evaluation techniques, the accuracy of the proposed method in modeling complex flows in curvilinear coordinate systems is confirmed, while the efficiency and simplicity of the method is maintained in the framework of the original Cartesian LBM-MRT.

2. The mathematical model

2.1. 2D Navier–Stokes equations transformed in a curvilinear coordinate system using complete transformation

The Navier–Stokes equations for 2D flow as defined in the Cartesian coordinate system (in horizontal x, y plane) are given by the continuity equation and the momentum equations in x and y directions:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0, \tag{1}$$

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho u v)}{\partial y} = S_x - \frac{\partial p}{\partial x} + \nu \frac{\partial^2(\rho u)}{\partial x^2} + \frac{\partial^2(\rho u)}{\partial y^2}, \tag{2}$$

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u v)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} = S_y - \frac{\partial p}{\partial y} + \nu \frac{\partial^2(\rho v)}{\partial x^2} + \frac{\partial^2(\rho v)}{\partial y^2}, \tag{3}$$

where t is time, x, y are the Cartesian coordinates, u, v are the Cartesian components of flow velocity, ρ is the density of the fluid, p is pressure, S_x and S_y are terms involving all additional influences (gravitation), while ν is the coefficient of kinematic viscosity.

Since domains of complex geometry are characterized mainly by arbitrary boundaries which do not coincide with the directions